

UNIFIED FINITE-ELEMENT METHOD DEVELOPMENT

B. T. Vu

NASA-Kennedy Space Center

A. M. Mobasher

Alabama A&M University

ABSTRACT

Finite element method (FEM) is traditionally recognized by its versatility for handling complicated geometries at a relative ease. This paper focuses on the development of a FEM solver for the Compressible Fluid Dynamics problems using the Mixed Explicit Implicit (MEI) method. The method utilizes object oriented program languages to create a unified environment for pre-processor, the main solver, and post processor. With this approach, the user can obtain the solution to the compressible Navier Stokes equations all in one environment, starting from the very raw data all the way to post-processing and analysis of the results. The success of this method is illustrated in several examples including compressible flow in laminar boundary layer shock interactions, ramjet, scramjet, and SSME nozzles. First, the grid is automatically generated upon selection of appropriate mesh parameters. On the same environment, the flow field parameters can also be specified before the flow solver is executed. Upon solution convergence, the results can be displayed on the same window. The procedure can be repeated until satisfactory results are achieved.

INTRODUCTION

Most programs for CFD analysis usually consist of environment in which the user must generate a separate input data that must be fed into the pre-processor. Then the results obtained by pre-processor must be checked by a graphic package to ensure the integrity of the resulting grid. Then CFD solver is run and a set of output data files are generated. The data files contains information about the flow field properties such as density, pressure, velocity, energy, temperature, etc. At the last stage, the results generated by the solver are viewed by a post processor. The results range from grid, density, pressure, Mach number contours or the surface properties such as skin-friction or pressure distribution. Then this process is repeated until a satisfactory solution is obtained. This process however appears to be inefficient and there exists a lot of "overhead" in the process that may be eliminated. The proposed development will integrate the three environments into one unified environment. The preprocessor, solver and post processor will be accessible simultaneously. This procedure will remove the overhead needed for accessing the data form one step to another.

METHOD OF SOLUTION

A robust FEM program that will aid in analysis and design related to high speed aerodynamics has been developed. The program will enable users to study and analyze flow around

complicated geometry such as ramjets, scramjets, internal flow in a space shuttle main engine or flow past conic shape objects with relative ease. The main feature of this program is its user-friendliness, in that the user will be able to access all input and output files in one unified environment. The Mix-Explicit Implicit algorithm in FEM will be utilized in the solution algorithm. The program will address the solution of in-viscid Navier Stokes equations and then this algorithm will be extended to cover the viscous cases. The methodology will also be extended to cover the viscous cases and the solution will be validated. The programs will be integrated into one unified environment using the object oriented programming languages such as Visual Basic, C++, Java, etc.

FORMULATIONS

The critical issues for the MEI code development are numerical scheme stability, discontinuity capturing and accuracy. The computer program will be developed based on the existing MEI numerical algorithm. The end product of this project will be a user-friendly environment for the solution of two-dimensional CFD problems using Finite Element Techniques. This code may be applied to investigate the high speed (supersonic to low hypersonic flow) fluid flow around or inside complicated geometry. In the following paragraph a brief description of the governing equations as well as the methodology used for the solution of these equations is described.

The Navier-Stokes system of equations characterizing the compressible viscous flows written in the conservation form is given by:

$$\frac{\partial U}{\partial t} + \frac{\partial F_i}{\partial x_i} + \frac{\partial G_i}{\partial x_i} = B \quad (1)$$

where U is the vector of unknowns (dependent variables), F is the convection flux vector, G is the diffusion flux vector and B is body force vector. Each of the variables in equation (1) are defined below:

$$U = \begin{pmatrix} \rho \\ \rho u \\ \rho v \\ \rho E \end{pmatrix}, F_x = \begin{pmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ uH \end{pmatrix}, F_y = \begin{pmatrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ vH \end{pmatrix} \quad (2)$$

$$G_x = -\frac{1}{\text{Re}} \begin{pmatrix} 0 \\ \tau_{xx} \\ \tau_{xy} \\ u\tau_{xx} + v\tau_{xy} + q_x \end{pmatrix}, G_y = -\frac{1}{\text{Re}} \begin{pmatrix} 0 \\ \tau_{yx} \\ \tau_{yy} \\ u\tau_{yx} + v\tau_{yy} + q_y \end{pmatrix}$$

Here ρ represents the density, u and v denote the x and y components of the velocity and p represents the pressure. The components of the stress tensor τ_{ij} are given by:

$$\tau_{xx} = \frac{2}{3}\mu_L(2\frac{\partial u}{\partial x} - \frac{\partial v}{\partial y}); \quad \tau_{xy} = \mu_L(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}); \quad \tau_{yy} = \frac{2}{3}\mu_L(2\frac{\partial v}{\partial y} - \frac{\partial u}{\partial x}) \quad (3)$$

The heat fluxes along the coordinate directions are defined as:

$$q_x = K\frac{\partial T}{\partial x}; \quad q_y = K\frac{\partial T}{\partial y} \quad (4)$$

and the total enthalpy, H , is given by:

$$H = (\rho E + p)$$

The next step in this process is to solve this system of equations. Although numerous algorithms have evolved for the solution of this system of equations, the methodology used here is the MEI algorithm using FEM.

The conservative form of governing equations (1) for compressible viscous flows is re written as:

$$\frac{\partial U}{\partial t} = -\frac{\partial F_i}{\partial x_i} - \frac{\partial G_i}{\partial x_i} - B \quad (5)$$

where U , F , G and B are the conservation variables, convection flux, diffusion flux, and source terms, respectively. For laminar flows in Cartesian coordinate system, and in the absence of body forces, the source term vector B is set to zero. Since $U=U(x,t)$, $F=F(U)$ and $G=G(U, U_{,j})$ we may write the following relations for F and G :

$$\begin{aligned} \frac{\partial F_i}{\partial x_i} &= a_i \frac{\partial U}{\partial x_i} \\ \frac{\partial G_i}{\partial x_i} &= b_i \frac{\partial U}{\partial x_i} + c_{ij} \frac{\partial U_{,j}}{\partial x_i} \end{aligned} \quad (6)$$

Where a_i , b_i and c_{ij} are the jacobians associated with the convection flux, diffusion flux and diffusion flux gradients, respectively.

U^{n+1} is expanded in Taylor series about U^n as follows:

$$U^{n+1} = U^n + \Delta t \frac{\partial U^{n+s_1}}{\partial t} + \frac{\partial^2 U^{n+s_2}}{\partial t^2} + O(\Delta t^3) \quad (7)$$

where s_1 and s_2 are the implicitness parameters associated with the first and second order derivatives with respect to time, respectively, with $0 < s_1 < 1$ and $0 < s_2 < 1$ and

$$\begin{aligned} \frac{\partial U^{n+s_1}}{\partial t} &= \frac{\partial U^n}{\partial t} + s_1 \frac{\partial \Delta U^{n+1}}{\partial t} \\ \frac{\partial^2 U^{n+s_2}}{\partial t^2} &= \frac{\partial^2 U^n}{\partial t^2} + s_2 \frac{\partial^2 \Delta U^{n+1}}{\partial t^2} \end{aligned} \quad (8)$$

Substituting equation (8) into equation (7) yields

$$\begin{aligned}
U^{n+1} = U^n + \Delta t \left(\frac{\partial U^n}{\partial t} + s_1 \frac{\partial \Delta U^{n+1}}{\partial t} \right) \\
+ \frac{\Delta t^2}{2} \left(\frac{\partial^2 U^n}{\partial t^2} + s_1 \frac{\partial^2 \Delta U^{n+1}}{\partial t^2} \right) + O(\Delta t^3)
\end{aligned} \tag{9}$$

The second derivative of U with respect to time may now be written in terms of the Jacobians defined by equation (6) as:

$$\begin{aligned}
\frac{\partial^2 U}{\partial t^2} = a_i \frac{\partial}{\partial x_i} \left(\frac{\partial F_j}{\partial x_j} + \frac{\partial G_j}{\partial x_j} - B^n \right) + b_i \frac{\partial}{\partial x_i} \left(\frac{\partial F_j}{\partial x_j} + \frac{\partial G_j}{\partial x_j} - B^n \right) \\
+ c_{ik} \frac{\partial^2}{\partial x_i \partial x_k} \left(\frac{\partial F_j}{\partial x_j} + \frac{\partial G_j}{\partial x_j} - B^n \right)
\end{aligned} \tag{10}$$

$$a_i = \frac{\partial F_i}{\partial U}; b_i = \frac{\partial G_i}{\partial U}; c_{ik} = \frac{\partial G_i}{\partial U_{,k}}$$

Substituting equations (5) and (10) into equation (8) and the resulting equations into equation (7) we arrive at U^{n+1} in the form,

$$\begin{aligned}
U^{n+1} = U^n - \Delta t \left[\left(\frac{\partial F_i}{\partial x_i} + \frac{\partial G_i}{\partial x_i} - B^n \right) + s_1 \left(\frac{\partial \Delta F_i^{n+1}}{\partial x_i} + \frac{\partial \Delta G_i^{n+1}}{\partial x_i} - \Delta B^{n+1} \right) \right] \\
+ \frac{1}{2} \Delta t^2 \left[(a_i + b_i) \frac{\partial}{\partial x_i} \left(\frac{\partial F_j}{\partial x_j} + \frac{\partial G_j}{\partial x_j} - B^n \right) + \frac{\partial B^n}{\partial t} \right] \\
+ s_2 \frac{1}{2} \Delta t^2 \left[(a_i + b_i) \frac{\partial}{\partial x_i} \left(\frac{\partial \Delta F_j^{n+1}}{\partial x_j} + \frac{\partial \Delta G_j^{n+1}}{\partial x_j} - \Delta B^{n+1} \right) \right]
\end{aligned} \tag{11}$$

In order to provide different amount of damping to the diffusion terms, two new implicitness parameters, s_3 and s_4 , are defined in the following manner:

$$\begin{aligned}
s_1 \Delta G &= s_3 \Delta G \\
s_2 \Delta G &= s_4 \Delta G
\end{aligned} \tag{12}$$

where s_3 and s_4 are associated with the first order and second order dissipative implicitness parameters, respectively. Substituting equation (12) into equation (11) and neglecting the third order spatial derivatives of the conservative variables associated with c_{jk} we arrive at the following relationship for the residual, R:

$$\begin{aligned}
R = & U^{n+1} - U^n + s_1 \Delta t \left(a_i \frac{\partial \Delta U^{n+1}}{\partial x_i} - \Delta B^{n+1} \right) \\
& + s_3 \Delta t \left(b_i \frac{\partial \Delta U^{n+1}}{\partial x_i} + c_{ij} \frac{\partial^2 \Delta U^{n+1}}{\partial x_i \partial x_j} \right) \\
& - \frac{1}{2} s_2 \Delta t^2 (a_i a_j + b_i a_j) \frac{\partial^2 \Delta U^{n+1}}{\partial x_i \partial x_j} \\
& - \frac{1}{2} s_4 \Delta t^2 (a_i b_j + b_i b_j) \frac{\partial^2 \Delta U^{n+1}}{\partial x_i \partial x_j} \\
& + \Delta t \left(\frac{\partial F_j}{\partial x_j} + \frac{\partial G_j}{\partial x_j} - B^n \right) \\
& - \frac{1}{2} \Delta t^2 \left[(a_i + b_i) \frac{\partial}{\partial x_i} \left(\frac{\partial F_j}{\partial x_j} + \frac{\partial G_j}{\partial x_j} - B^n \right) + \frac{\partial B^n}{\partial t} \right] + O(\Delta t^3)
\end{aligned} \tag{13}$$

All the Jacobians, a_i , b_i and c_{ij} in equation (13) are assumed constant in space within each time step. Equation (13) is the basis for the galerkin finite element formulation. The galerkin finite element formulation of the above relation is obtained by taking the inner product of the residual R and the test function Φ_α and integrating over the domain, Ω :

$$\int_{\Omega} R(U, F_i, G_i) d\Omega = 0 \tag{14- a}$$

where,

$$\begin{aligned}
U(x, t) &= \Phi_\alpha(x) U_\alpha(t) \\
F(x, t) &= \Phi_\alpha(x) F_\alpha(t) \\
G(x, t) &= \Phi_\alpha(x) G_\alpha(t)
\end{aligned} \tag{14 b-d}$$

The compact form of this formulation is given by the following relationship:

$$(A_{\alpha\beta} \delta_{rs} + B_{\alpha\beta rs}) \Delta U_{\beta s}^{n+1} + N_{\alpha r}^{n+1} = H_{\alpha r}^n + N_{\alpha r}^n \tag{15}$$

here α, β are the indices associated with the global node and r, s are the indices associated with the individual equations. Each of the matrices $A_{\alpha\beta}$, $B_{\alpha\beta rs}$, $N_{\alpha r}^{n+1}$, $N_{\alpha r}^n$ and $H_{\alpha r}^n$ are defined as follows:

$$\begin{aligned}
A_{\alpha\beta} &= \int_{\Omega} \Phi_{\alpha} \Phi_{\beta} d\Omega \\
B_{\alpha\beta rs} &= \int_{\Omega} \{ \Delta t [-s_1 a_{jrs} \Phi_{\alpha,j} \Phi_{\beta} \\
&\quad - s_3 (b_{jrs} \Phi_{\alpha,j} \Phi_{\beta} + c_{jirs} \Phi_{\alpha,j} \Phi_{\beta,i})] \\
&\quad + \frac{1}{2} \Delta t^2 [s_2 (a_{jr q} a_{is q} + b_{jr q} a_{is q}) \Phi_{\alpha,j} \Phi_{\beta,i} \\
&\quad + s_4 (a_{jr q} b_{is q} + b_{jr q} b_{is q}) \Phi_{\alpha,j} \Phi_{\beta,i}] \} d\Omega \\
N_{\alpha}^{n+1} &= \int_{\Gamma} \{ \Delta t [-s_1 a_{jrs} \Phi_{\alpha}^* \Phi_{\beta}^* - s_3 (b_{jrs} \Phi_{\alpha}^* \Phi_{\beta}^* + c_{jirs} \Phi_{\alpha}^* \Phi_{\beta,i}^*)] \\
&\quad + \frac{1}{2} \Delta t^2 [s_2 (a_{jr q} a_{is q} + b_{jr q} a_{is q}) \Phi_{\alpha}^* \Phi_{\beta,i}^* \\
&\quad + s_4 (a_{jr q} b_{is q} + b_{jr q} b_{is q}) \Phi_{\alpha}^* \Phi_{\beta,i}^*] \} n_j \Delta U_{\beta s}^{n+1} d\Gamma \\
N_{\alpha}^n &= - \int_{\Gamma} [\Delta t \Phi_{\alpha}^* \Phi_{\beta}^* (F_{\beta r}^n + G_{\beta r}^n) - \frac{1}{2} \Delta t (a_{jrs} + b_{jrs}) \Phi_{\alpha}^* \Phi_{\beta,i}^* (F_{is}^n + G_{is}^n)] n_j d\Omega \\
H_{\alpha}^n &= \int_{\Omega} \{ \Delta t [\Phi_{\alpha,j} \Phi_{\beta} (F_{\beta r}^n + G_{\beta r}^n)] \\
&\quad - \frac{1}{2} \Delta t^2 (a_{jrs} + b_{jrs}) [\Phi_{\alpha,j} \Phi_{\beta,i} (F_{\beta s}^n + G_{\beta s}^n)] \} d\Omega
\end{aligned} \tag{16}$$

Here the source terms associated with B are neglected. In this formulation Φ indicates the interpolating functions associated with the flux terms inside the domain and the '*' denotes the interpolating function associated with the terms in the boundary.

ACOUSTIC MODELING

The wave equation that describes sound propagating in free air is governed by:

$$-\Delta p + \frac{1}{c^2} \left(\frac{\partial^2 p}{\partial t^2} \right) = 0$$

which can be linearized at the outside surface by the Kirchhoff formulation:

$$\nabla^2 p' - \frac{1}{c_{\infty}^2} \left(\frac{\partial}{\partial t} + u_{\infty} \frac{\partial}{\partial x} \right)^2 p' = 0$$

which reduces to a simple wave equation in the special case of stationary sources ($u_{\infty} = 0$). A solution to the pressure field can be expressed by the surface integrals as (Morino and Tseng 1990, Lyrantzis & Mankbadi 1996)

$$p'(x, y, z, t) = -\frac{1}{4\pi} \int_{\Gamma} \left[\frac{p'}{r_0^2} \frac{\partial r_0}{\partial n_0} + \frac{1}{r_0} \frac{\partial p'}{\partial n_0} + \frac{1}{c_{\infty} r_0 \beta^2} \frac{\partial p'}{\partial t} \left(\frac{\partial r_0}{\partial n_0} - M_{\infty} \frac{\partial x'_0}{\partial n_0} \right) \right]_{\tau} dS_0$$

where

$$r = \left\{ (x - x')^2 + \beta^2 \left[(y - y')^2 + (z - z')^2 \right] \right\}^2$$
$$\tau = \frac{[r_0 - M_\infty (x - x')]}{c_\infty \beta^2}, \quad \beta = (1 - M_\infty^2)^{1/2}$$

In the above equations, M_∞ is the freestream Mach number, c_∞ the speed of sound in the freestream, the prime a point on the Kirchhoff surface, τ is the retarded time $\tau = t - t'$, n_0 the outward normal to the Kirchhoff surface S_0 . Thus the pressure at any instant in the region outside the Kirchhoff surface can be expressed in terms of the information prescribed on the Kirchhoff surface. The required data on pressure, and its normal and temporal derivatives on the Kirchhoff surface are taken from the CFD solution.

USER-FRIENDLY ENVIRONMENT

The next task is to develop a user friendly environment so that input data, may be generated, program may run on a PC based computer and the graphical results may be viewed, and analyzed immediately. This aspect of the research is viewed an important aspect, because the computational costs associated with the calculations is drastically reduced as compared with the solutions obtained on the supercomputers.

BENCHMARK CASES

The success of the development has been demonstrated through several applications. Figure 1 shows the computational grid for laminar shock wave boundary layer shock interaction. As the computation progresses, a shock wave is formed and its interaction with the boundary layer can be detected, as shown in Figure 2. In another application, the grid is also clustered near the boundary surface, as for the ramjet case (Figure 3). Upon solution convergence, the density and pressure contours can be plotted in Figures 4 and 5, respectively. The area of interest can be zoomed in, as shown in Figure 6, to focus on the position where shock waves interact with the boundary layer. Here, the recirculation zone is clearly visible. Figures 7 and 8 show the grid and solution contours inside a rocket nozzle. Figure 9 is the computational grid generated around a scramjet nozzle. Throughout these examples, the user can generate the grid, specify the flow condition, solve the flow field, and visualize the solution – all in one environment. The process can be repeated until a satisfactory solution has been achieved.

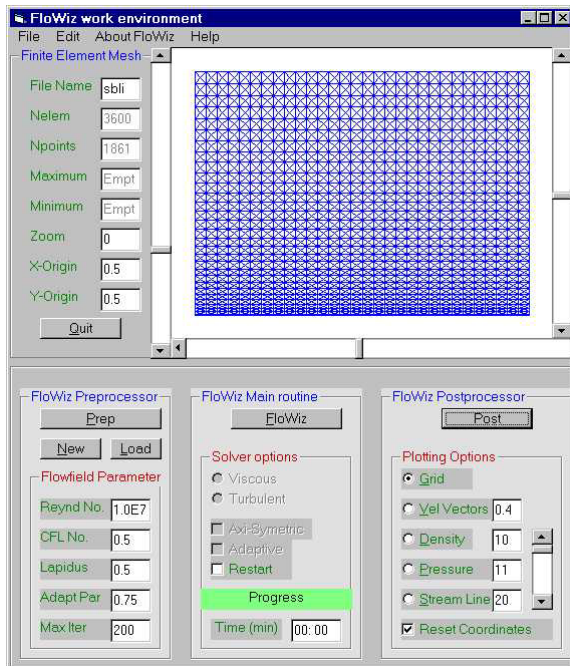


Figure 1. Grid for laminar shock wave boundary layer interaction

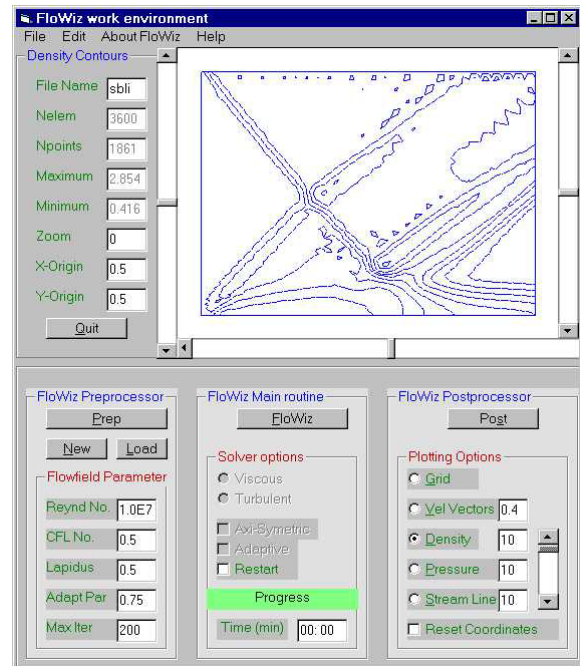


Figure 2. Density contours for the shock wave boundary layer interaction

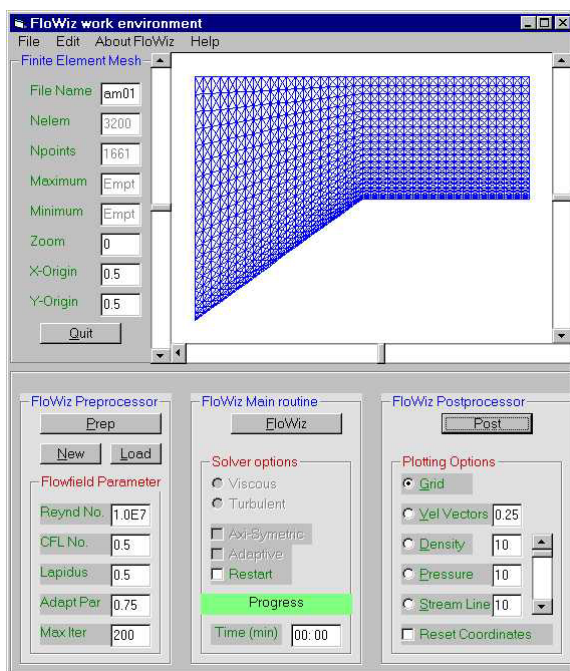


Figure 3. Grid for the ramjet

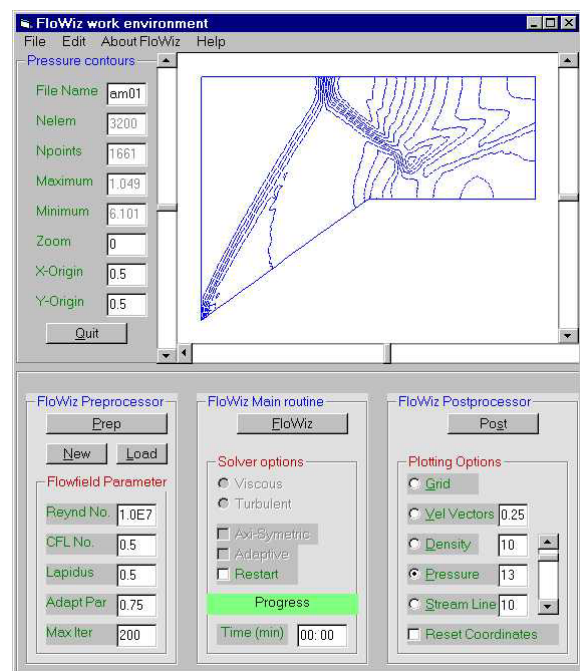


Figure 4. Pressure contours for the ramjet

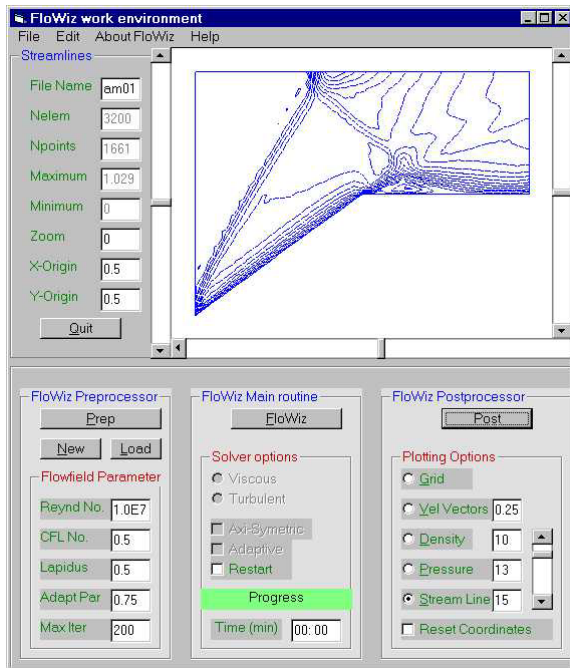


Figure 5. Stream line contours for ramjet.

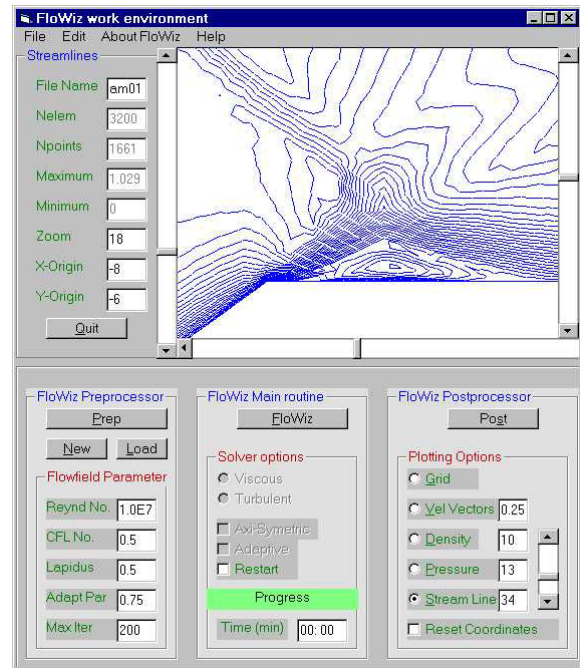


Figure 6. Close up view of the stream line contours

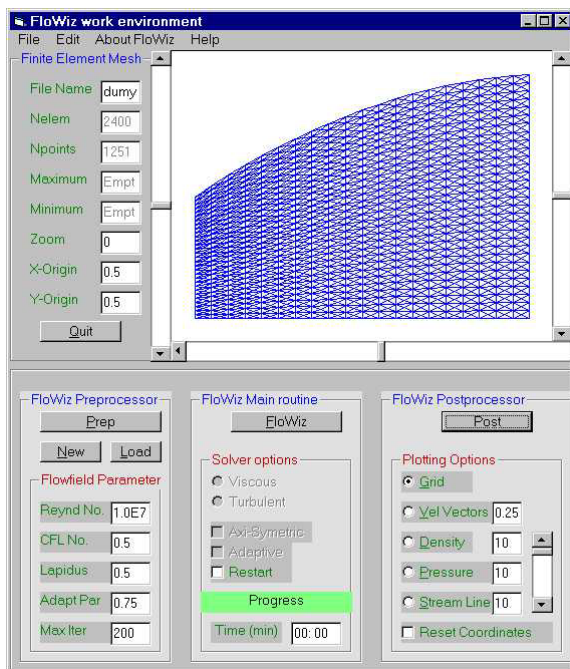


Figure 7. Mesh for a nozzle

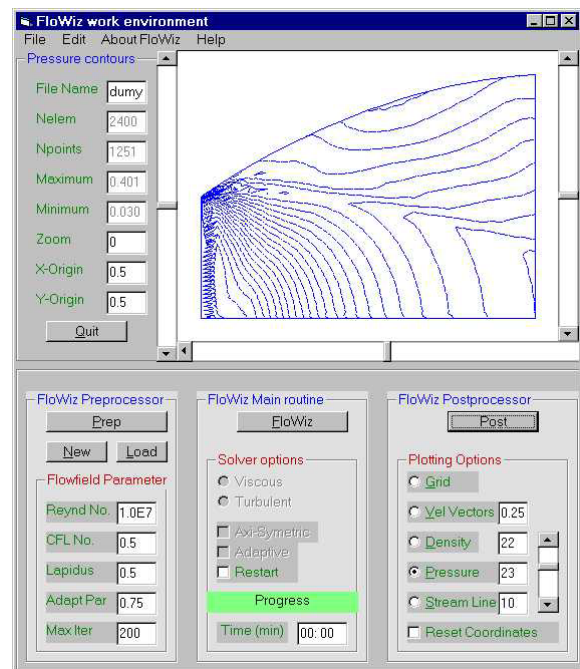


Figure 8. Pressure contours for flow inside a nozzle

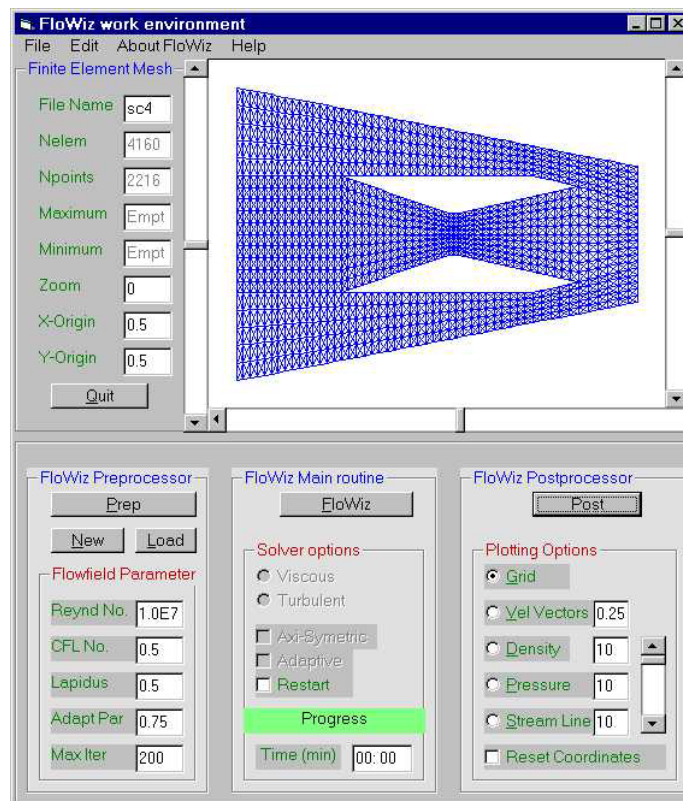


Figure 9. Geometry of scramjet.

CONCLUSIONS

A unified FEM has been developed for solving compressible fluid dynamics problems. The development is aimed to allow the achievement of quick and accurate solution using MEI scheme. In addition, the goal of this unified development is to integrate pre-processing, flow solver, and post-processing into one environment, thus alleviate many “overheads” in solving CFD problems.

At the time of this publication, the code has been successfully modified to include a solution-adaptive procedure in an effort to improve the solution efficiency. Other ongoing developments include but are not limited to acoustic modeling, turbulence modeling, and 3-D extension.

ACKNOWLEDGEMENTS

The authors would like to acknowledge the continual support from University Program office, Testbed Technology Program, and Equal Opportunity office at NASA Kennedy Space Center. Thanks are also due to the research institute and R&D office at Alabama A&M University for initiating the university grant that led to this development.

REFERENCES

1. Lecture notes. Professor T.J. Chung. University of Alabama in Huntsville.
2. Amir A. Mobasher, “Effects of compressibility and Dilatation Dissipation on Shock-Wave Turbulent Boundary Layer Interactions,” PhD Dissertation, University of Alabama in Huntsville, 1994.
3. Amir A. Mobasher, T.J. Chung, “Effects of Compressibility in Shock wave turbulent Boundary Layer interactions,” AIAA Paper, 96-2154.
4. Mobasher, A, Bernstein, E., and Park, Y. “Application of MEI Finite Element to Store Separation”, Presented at Symposium on Finite Element Applications in Fluid Dynamics, July 7-11, 1996, San Diego, CA.
5. Acoustic Loads Generated by The Propulsion System- NASA SP-8072